Getting Started with MapleSim Insight

Getting Started with MapleSim Insight

Copyright

Maplesoft, Maple, and MapleSim are all trademarks of Waterloo Maple Inc.

This document was produced using Maple and DocBook.

Introduction

MapleSim Insight lets you see live simulation results and 3-D visualizations during the simulation of any MapleSim-generated FMU. MapleSim Insight works with any simulation tool that supports compiled FMUs (Functional Mock-Up Units). MapleSim Insight provides 3-D visualizations for quick visual feedback, and 2-D plots for detailed testing and debugging.

Basics of using MapleSim Insight:

• Model your system in MapleSim.

• Generate a compiled FMU in the **MapleSim FMI Connector** with the **Include MapleSim Insight data** option selected.

• Run the FMU. For Co-Simulation 2.0 FMUs, you can simulate directly in MapleSim Insight. Or, you can simulate in MapleSim or in any tool that can import and simulate an FMU. If not already open, MapleSim Insight launches automatically.

• MapleSim Insight shows continuously updated simulation data plots and an animated 3-D view while the FMU is running.

• Controls in MapleSim Insight allow you to alternate between steaming live data and reviewing previously captured data.

• Record and save streamed data to file to review at a later time.

Requirements

Generating an FMU for use with MapleSim Insight requires MapleSim and the MapleSim FMI Connector, release 2020 or later.

Using MapleSim Insight requires the generated FMU, MapleSim Insight, and a tool capable of simulating compiled FMUs. (You do not need MapleSim to run MapleSim Insight.)

Simulating through MapleSim Insight works with Co-Simulation 2.0 FMUs. This feature only works with FMUs generated in release 2020.2 or later.

Using Help

Tip: To search for help on a specific topic, in the Help window, press **Ctrl** + **F** and then enter a search term.

Generating an FMU for Use with MapleSim Insight

MapleSim Insight works with FMUs that are exported using the MapleSim FMI Connector. This step requires MapleSim and the MapleSim FMI Connector.

1. Build the model in MapleSim.

2. Use the FMU Generation app to create the FMU file.

    a. Select the subsystem to export and specify desired settings.

    b. Ensure that you have selected the **Include MapleSim Insight data** option under Export Options. (This option is selected by default.)

    c. Click **Generate FMU Archive**. The .fmu zip file is created and saved in the target directory.

Launching MapleSim Insight

MapleSim Insight can be launched in several ways.

- Launch MapleSim Insight directly.

- Simulate an FMU in any tool that can simulate an FMU. When the FMU starts running, MapleSim Insight launches.

Tip: If you launch MapleSim Insight directly, and then select an FMU to open, you can set up the 3-D and plot windows as desired before you begin the simulation. When you then start the simulation and data is received, the windows populate immediately and it is easy to view the data.

Notes about display:

- If you open an fmu that has no 3D components, then Insight should only display the traces window (and no 3D view).

- If you open an fmu that has no fmu outputs. then Insight should only display the 3D window (no traces window).

- If you open an fmu that has no 3D or no fmu outputs, then Insight should show no 3D and no traces window.

Note about language: By default, MapleSim Insight starts in the language based on your system's locale setting. For example, if you are using an English system, MapleSim Insight starts in English. If you are using a Japanese system, MapleSim Insight starts in Japanese. To instead specify the locale for the application, launch MapleSim Insight using the following command-line argument:

```
>insight.exe -locale en //run in English
```
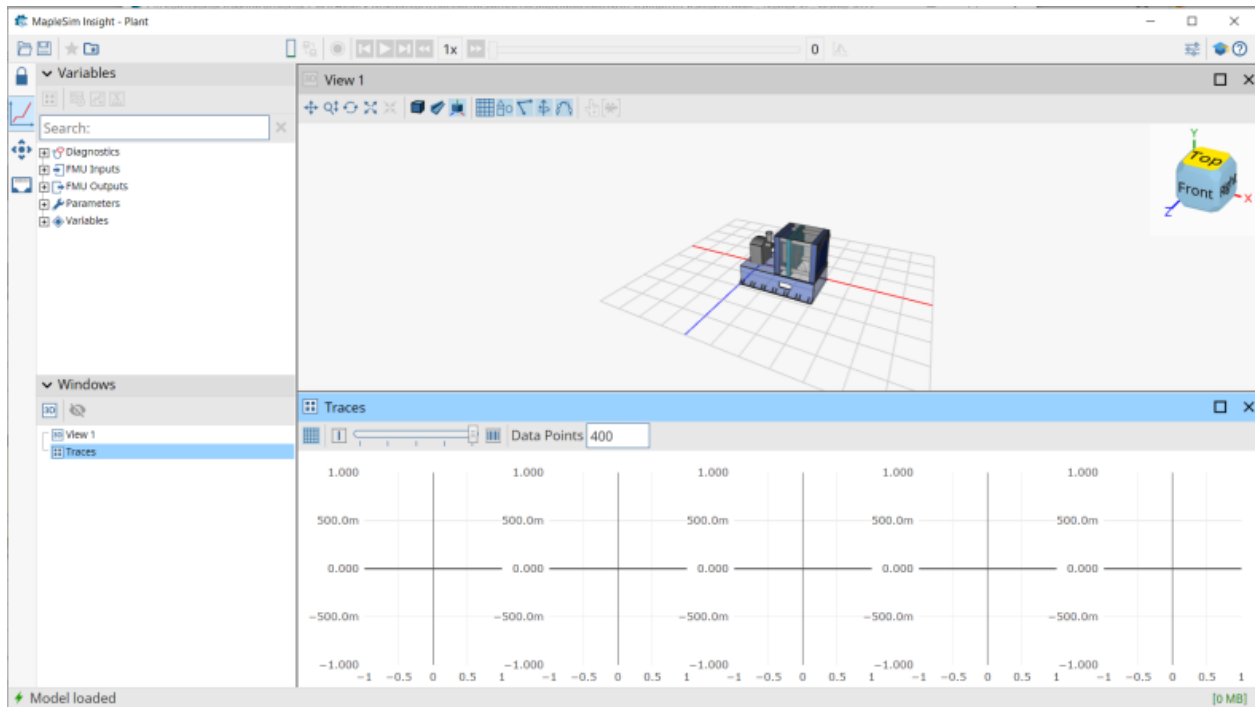
```
>insight.exe -locale ja //run in Japanese
```
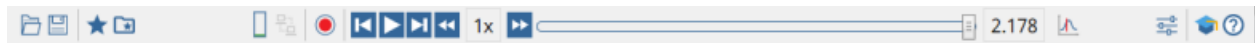
Getting Started with MapleSim Insight

The MapleSim Insight Window

The MapleSim Insight window contains the following panes and components:
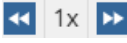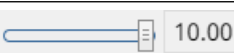
- The main toolbar at the top
- The left pane containing tabs for working with the simulation results and for running simulations
- The visualization component in the main area, containing the 3-D visualization windows and 2-D plot windows
- The status bar at the bottom displaying system information

The following icons are found in the Main toolbar:

Icon	Description
	Open data source.
	Save.
	Save Latest Results. Use this button to save and rename recently run simulation results.
	Manage Results. Use this button to access the Results Management window.
	Data Storage Rate indicator. When the bar is empty or low (only showing green) this means that the computer is able to keep up with the data being sent to Insight. As this bar rises and turns orange or eventually red, then this indicates that Insight is having problems keeping up processing the incoming simulation data. As a result, the internal cache

	of the computer buffers really quickly. These buffers have a limit in terms of how big they are able to grow, so if the simulation doesn't stop before the buffer gets 'full' then the system will crash. To prevent this from happening. Insight will display this indicator growing from green, orange to red. When the indicator reaches the top, Insight will automatically disconnect from the running simulation to prevent it from crashing. This means that the simulation can continue to run, but now you are no longer recording the data that it is spitting out. You can tune their parameters and try to reconnect. The parameters to tune are mentioned in the error dialog that pops up when we disconnect.
	Pause/resume connection to streaming data. Playback buttons are not available while connected to streaming data.
	Record Video
	Go to start.
	Start/Stop playback.
	Go to end.
1x	Playback speed. Increase or decrease the speed at which the animation plays.
10.00	Timeline. Indicates the current position in playback.
	Show/hide time indicator. Indicates the current time in the plots. See *Working with Plot Windows (page 11)*.
	Open Application Settings.
	Tutorial Mode toggle button.
	Help.

When you save simulation data, it can be saved as a .simData file or an .fmu file. You can later open saved .simData files. The saved simulation data includes data from a window in time. You can control the time duration of this window by adjusting how much data is stored. See Max Buffer Size in *Application Settings (page 17)*.

However, if you intend on running the simulation in another tool, consider saving the simulation data as an .fmu file to preserve window layout information.

The status bar at the bottom right corner of the MapleSim Insight window displays how much memory has been used to store the collected data. The value is shown in green if it is < 75% of the max buffer size. If it reaches 75% of the max buffer size, the text color is orange. If it reaches 90% of the max buffer size, the text color is red. This memory usage information is also available in Application Settings.

At any point while the simulation is running, you can pause the connection using the **Pause/Resume Connection** button ( ) to analyze the data and then resume the connection to pick back up real-time view. Any data accumulated while the connection is paused will be discarded when you resume the connection.

For information on controlling the connection options, see *Application Settings (page 17)*.

Getting Help

You can find help for a variety of topics related to MapleSim Insight using this guide, or through the self-guided tutorial.

Using the MapleSim Insight Tutorial

The first time you launch Insight:

1. By default the first time you launch Insight, **Show tutorial on start** is checked. Uncheck this box to stop the Welcome to Insight window from opening whenever you launch Insight.

2. To begin the tutorial, click **Next**.

Note: The Welcome to Insight window is launched, only when you open Insight directly. If you launch Insight indirectly (for example, double-clicking an .fmu file in Windows) the Welcome to Insight window will not launch.

Using the tutorial after your initial launch of Insight:

1. Click Tutorial Mode ( ). This opens the Welcome to Insight window.

2. If you have an FMU model already open in Insight, the Welcome Insight window will be identical to the one displayed for your initial Insight launch. If you do not have a suitable .fmu file, use tutorial.fmu found in `$BIN/resource/`. Where `$BIN` is the bin subdirectory of your MapleSim Insight installation.

3. Once you have opened your FMU model, click **Next** to begin the tutorial.

Note: You can change the focal point (point that you are rotating about) of the model by selecting your new new focal point in the 3-D Scene, then pressing **F**.

Working with the Left Pane

The left pane of the MapleSim Insight window contains tabs for working with the simulation results and for running simulations.

- **Simulation Results** ( ): Access tools for managing your 3-D visualization and plot windows.

- **Simulate Co-Simulation FMU** ( ): For Co-Simulation FMUs, access simulation controls for simulating the FMU directly from MapleSim Insight.

Click on the desired button to view that tab.

Simulation Results	Simulate CoSim FMU	EtherNet/IP Adapter Settings
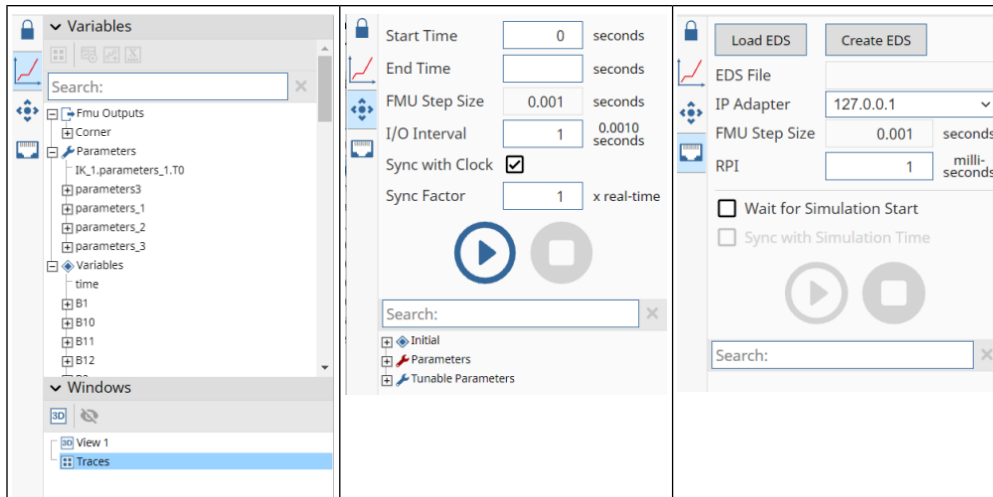
You can collapse the left pane to allocate more room for the visualization component in the main pane.

To collapse the pane:

• Click **Show/Hide** (🔒) for the pane.

To access the contents of the collapsed pane, select one of its tabs. The pane stays expanded until you click somewhere outside of that tab.

To expand the collapsed pane:

• Click **Show/Hide** (🔓) for the pane. The pane now stays open.

Simulating a Co-Simulation FMU in MapleSim Insight

In the Simulate Co-Simulation FMU tab (⬌) of the left pane, you can run a simulation directly from within MapleSim Insight. This feature is only available for Co-Simulation FMUs version 2.0 created using MapleSim FMI Connector 2020.2 or later.
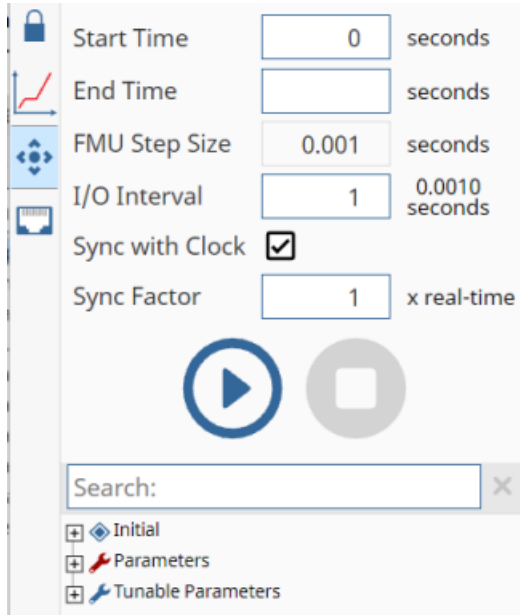
Start Time. Default value is 0.

End Time. The default is no value set, meaning it never stops.

FMU Step Size. This is determined when the FMU is generated and is not editable.

I/O Interval. How often input/output is communicated.

Sync with Clock. When selected and the sync factor is 1, attempts to run simulation in real time. Note: The ability to sync to clock can be limited by processing power.

Sync factor. This option is only available when sync with clock is selected. Determines the sync factor of the simulation: setting to 10 means 10 times faster than real time, and setting to 0.5 means half time. Default is 1.

Note: The running speed of the simulation, with sync factor applied can be seen in the status bar of the MapleSim Insight window.
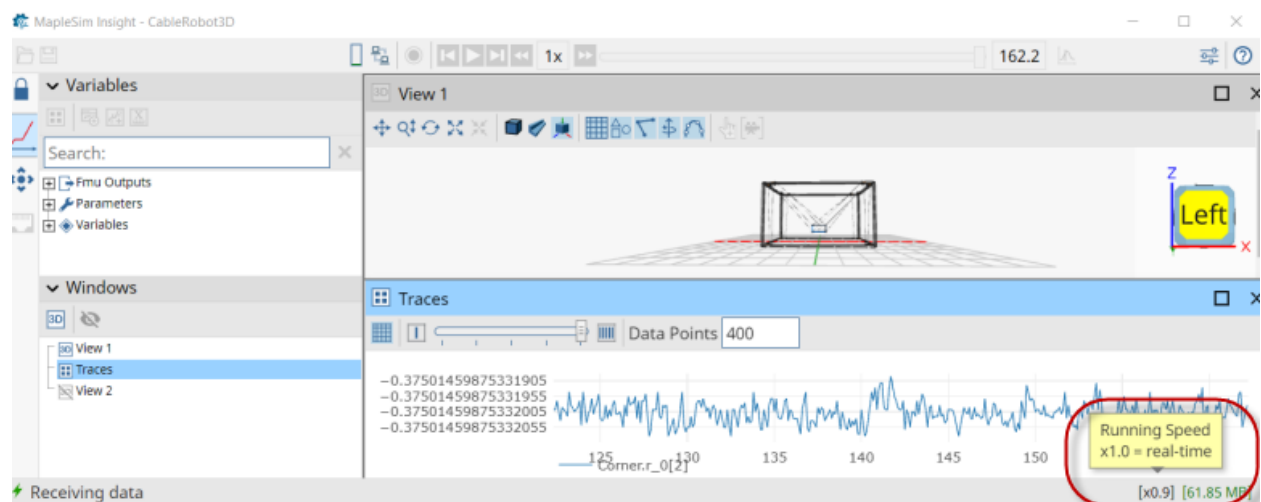
Before running a simulation, you can modify parameters. Use the tree view to browse for the parameters you want. Parameters are categorized into initial, inputs, parameters (nontunable) and tunable parameters. The red wrench (🔧)

indicates nontunable parameters and the blue wrench ( ) indicates the tunable parameters. Alternatively, use the search box to find parameters.

Note:

- You can use wildcard characters (*) in the search box.

- Once you have found the variables you want to plot from the search results, you can use the CTRL and Shift keys to select multiple variables to plot.

To modify a value before running a simulation:

- Select a variable.

- Double-click and enter new value. The modified value is shown in green to indicate it was changed.

To reset to the default value, delete the new value and press enter. It returns to the original value.

Tunable parameters and inputs can be modified on the fly while a simulation is running. Click pause ( ), change

values as needed, and continue simulation ( ).

Note: Any tuning changes you make here are only temporary. Your changes are only used for the current session of MapleSim Insight or until you load another FMU.

Simulating through Another Tool

You can simulate using another simulation tool that supports compiled FMUs (Model Exchange or Co-Simulation FMUs) and view live simulation results in MapleSim Insight.

- Run the FMU in MapleSim or in any tool that can import and simulate an FMU. If not already open, MapleSim Insight launches automatically.

- MapleSim Insight shows continuously updated simulation data plots and an animated 3-D view while the FMU is running.

- Controls in MapleSim Insight allow you to alternate between steaming live data and reviewing previously captured data. During simulation, you can pause the connection using the **Pause/Resume Connection** button ( ) to analyze the data and then resume the connection to pick back up real-time view.

Best Practices: You should only control the FMU from one source. Do not try to simulate it from within Insight and from another tool at the same time.

Working with 3-D Visualizations

In the 3-D window, the following icons are found in the **Toolbar**:

Icon	Description
	Pan the view.
	Zoom in or out.
	Rotate.
	View the entire model, or if objects are selected, fill the view with the selected objects.

✖	Focus view on the selected object.
⬛	Rendering Mode: Shaded plus wireframe
⬛	Rendering Mode: Wireframe
⬛	Rendering Mode: Shaded
◆	Toggle Camera Type: Perspective view
◆	Toggle Camera Type: Orthographic view
⬛	Show/hide orientation.
▦	Show/hide gridlines.
△□○	Show/hide shapes.
⌐	Show/hide implicit geometry.
⇕	Show/hide arrows.
⌒	Show/hide trace lines.
✋	Select tracking target.
🎥	Toggle camera tracking.

The orientation box displays the orientation of the visualization.
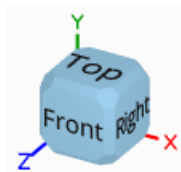
You can change the Up Direction of a 3-D Window by right-clicking on the orientation box and selecting which axis should be up.

By default, the positive Y axis is up, and the grid (when gridlines are visible) lies on the XZ plane. When the up direction is changed, the grid is changed as well so it lies perpendicular to the up direction. To change the plane that the grid spans, see the 3-D Settings described in the next section. You can also set the up direction and show/hide gridlines in this dialog.

Customize 3-D Settings for a 3-D Window

To access the 3-D Settings for a 3-D window, right-click on the window and select **3-D Settings**.
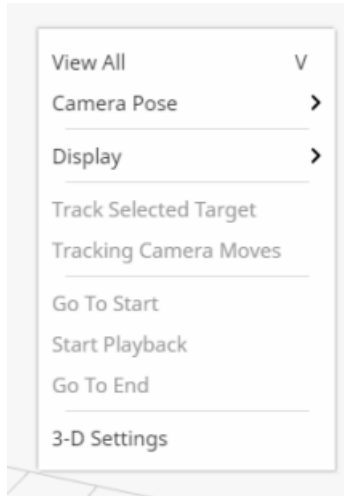
Here you can customize settings for the grid and 3-D view.

Other settings for 3-D windows are found in the Application Settings dialog. These settings are shared among all 3-D windows. See *Application Settings (page 17)*.

Camera Pose Submenu

- **Save.** Click Save to store the position and orientation of the camera.

- **Restore.** Click Restore to move back to the last-saved position and orientation of the camera.

Grid

- Set the **Up Direction**. (By default, it is the positive Y axis.)
  You can also change the Up Direction of a 3-D Window by right-clicking on the orientation box and selecting which axis should be up.

- **Visible**. Use this to show/hide gridlines. You can also set the plane on which the grid is displayed. (By default, it is the XZ Plane.)

- Set the grid **Spacing**. Note: This setting is shared among all 3-D windows, and can also be accessed in the Application Settings dialog.

Tip: If the model animates over a large enough area to move outside the bounds of the current grid extents, you can fix this by toggling the grid visibility off and back on. When gridlines are shown again, new grid extents are computed, better reflecting the current state of the model.

3-D View

- **Clipping Planes**. For expansive models, the near/far planes settings are used to determine which objects are rendered and which are ignored. By default, this is set to Auto. If you clear the Auto check box, you can manually set values for Near and Far.

Working with Traces and Plots

As of Maple Insight version 2021.1, you can visualize your FMU simulation results using traces.

Traces are similar to plots in that they are a tool you can use to visualize your simulation results. They differ in the data they display.

Where plots display the complete data set for the entire duration of the simulation, traces display a snapshot of data generated during a specific interval. This interval and the generated data, are continually updated.

This makes it possible to load more complex models, run simulations faster, and run simulations without lag.

Notes:

- There is a limit of ten traces allowed to ensure streaming performance.

- There is no limit on the number of plots.

Working with the Trace Window

The Trace window includes a plot for each input and output variable.
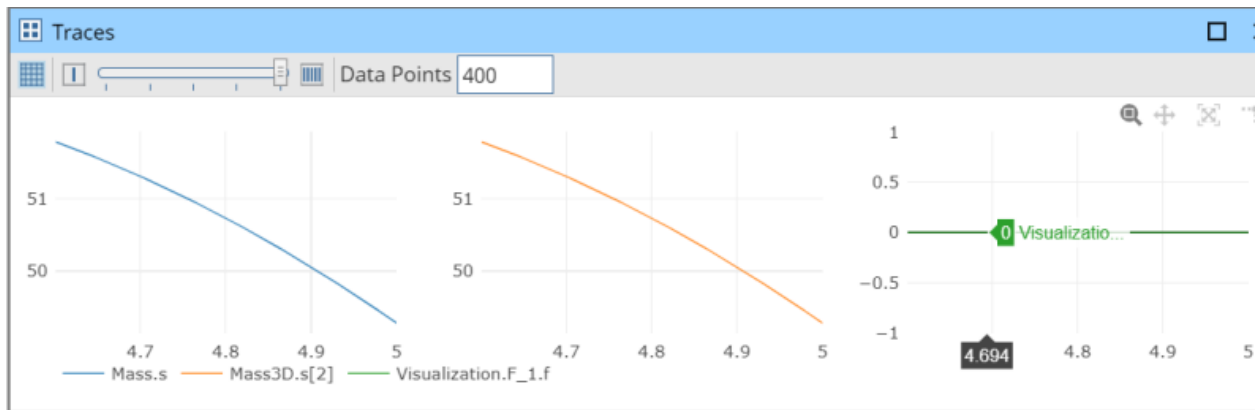
In the Traces window, the following icons are found in the **Toolbar**:

Icon	Description
	Show/hide gridlines for all the trace plots.
	1 Column of Plots. Change the **Traces** window to display the trace plot(s) in a single column.
	Number of plot columns. Select the number of plot columns (1-5) in the trace plot window layout by using the slider.
	5 Columns of Plots. Change the **Traces** window to display the trace plots in five columns.
Data Points 400	**Data Points**. The value in this field indicates the number of points displayed and stored in each trace plot. The default value is 400.

Hiding and Removing Trace Plots

To hide a trace trace plot, click on the legend name. The trace plot will disappear and the trace plot legend will be grayed out.

To make the trace plot reappear, click on the grayed out legend name.

To permanently delete a trace plot or to delete all trace plots:

1. Right click anywhere in the **Traces** window to display the context menu.

2. From the context menu, select either **Delete All Traces** or **Delete Trace**.

3. If you selected **Delete Trace** in the previous step, select the variable you want to remove from the submenu.

4. The trace plot is removed.
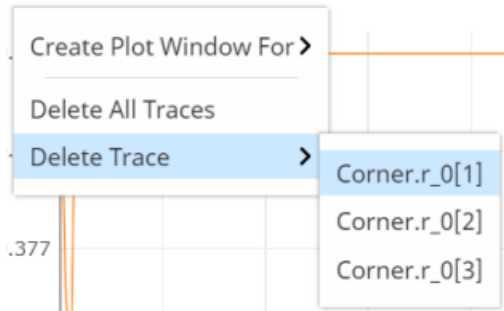
Working with an Individual Trace

When you hover over an individual plot inside a trace window, a pop-up toolbar is available.
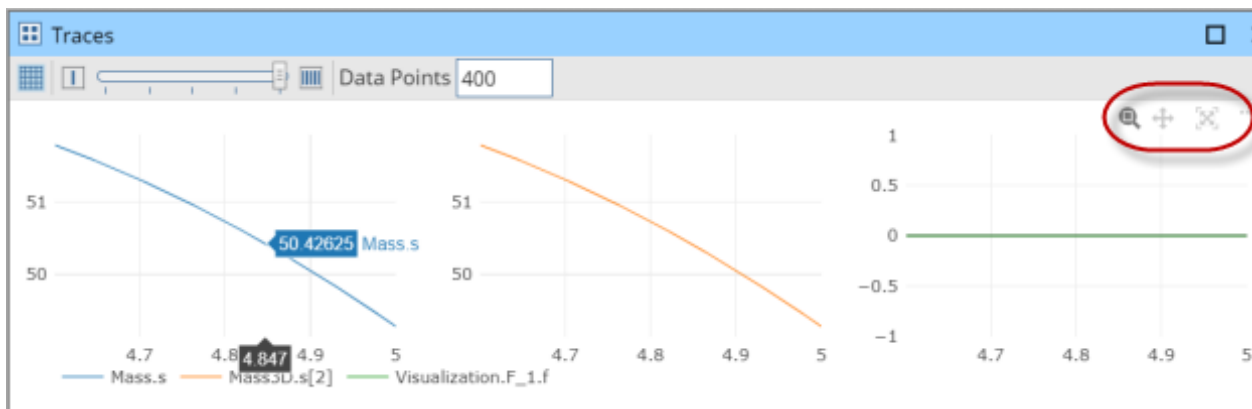
The following tools are displayed in the plot toolbar.

Control	Description
🔍	Zoom.
✛	Pan.
⤢	Autoscale. Resets view.

Use the pop-up toolbar to zoom, pan, zoom in and out, and autoscale.

When in **Zoom** mode (🔍), drag the pointer across the plot region to select a specific region to zoom in on. The selected region fills the plotting area. To return a trace to the original view, click **Autoscale** (⤢). Alternatively, double click in a trace to return to the original view.

When in **Zoom** mode (🔍), you can pan by holding **Shift** and dragging the pointer across the trace. If you drag along the x- or y-axis, you can pan only right/left or up/down.

Creating a Plot from a Trace

You can create plots for variable data using the trace context menus.

To create a plot:

1. Right-click anywhere in the **Traces** window.

2. From the context menu select **Create Plot Window For**.

3. From the submenu, select the variable that you want to plot.

A plot window is created for the variable you want to plot.

Working with Plot Windows

The Plots window includes a plot for each input and output variable.
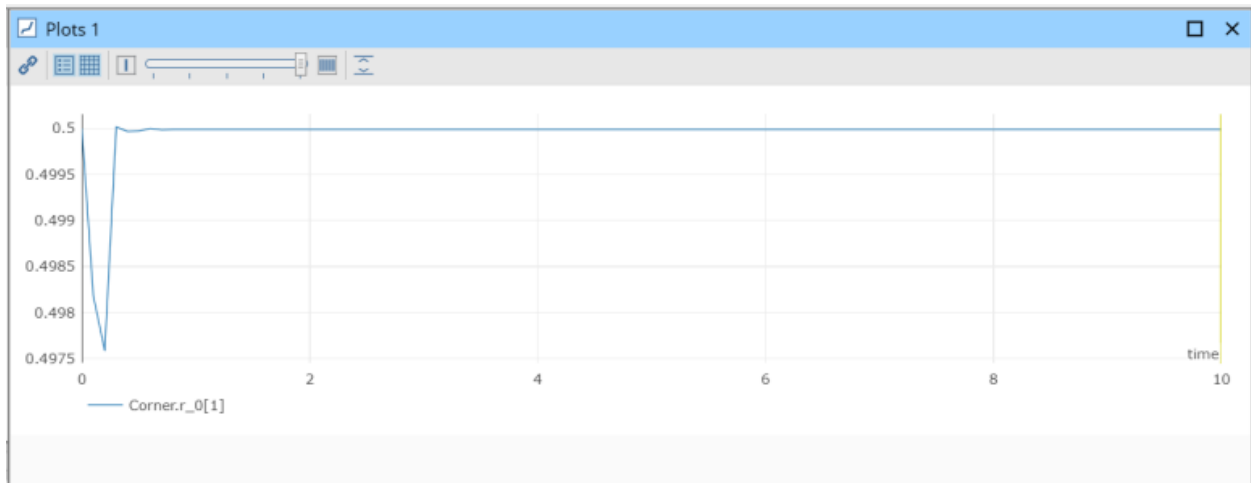
In the plot window, the following icons are found in the **Toolbar**:

Icon	Description
	Link plots. Link the plots in a plot window actions performed on one plot are replicated on all plots.
	Show/hide legend.
	Show/hide gridlines.
	1 Column of Plots. Change the **Plots** window to display the plot(s) in a single column.
	Number of plot columns. Select the number of plot columns (1-5) in the plot window layout by using the slider.
	5 Columns of Plots. Change the **Plots** window to display the plots in five columns.
	Fit plots to plot window. Adjust the height of all graphs so that they fit in the current height of the plot window.

When link plots ( ) is selected, most actions performed on one plot are replicated on the other plots. This includes changing the x-axis range, zoom all, show/hide legend, and toggle gridlines.

You can reorder plots in a plot window. To move a plot, hold **Ctrl** and drag one plot on top of another plot. The selected plot is moved to the new location.

Working with an Individual Plot

When you hover over an individual plot inside a plot window, a pop-up toolbar is available.
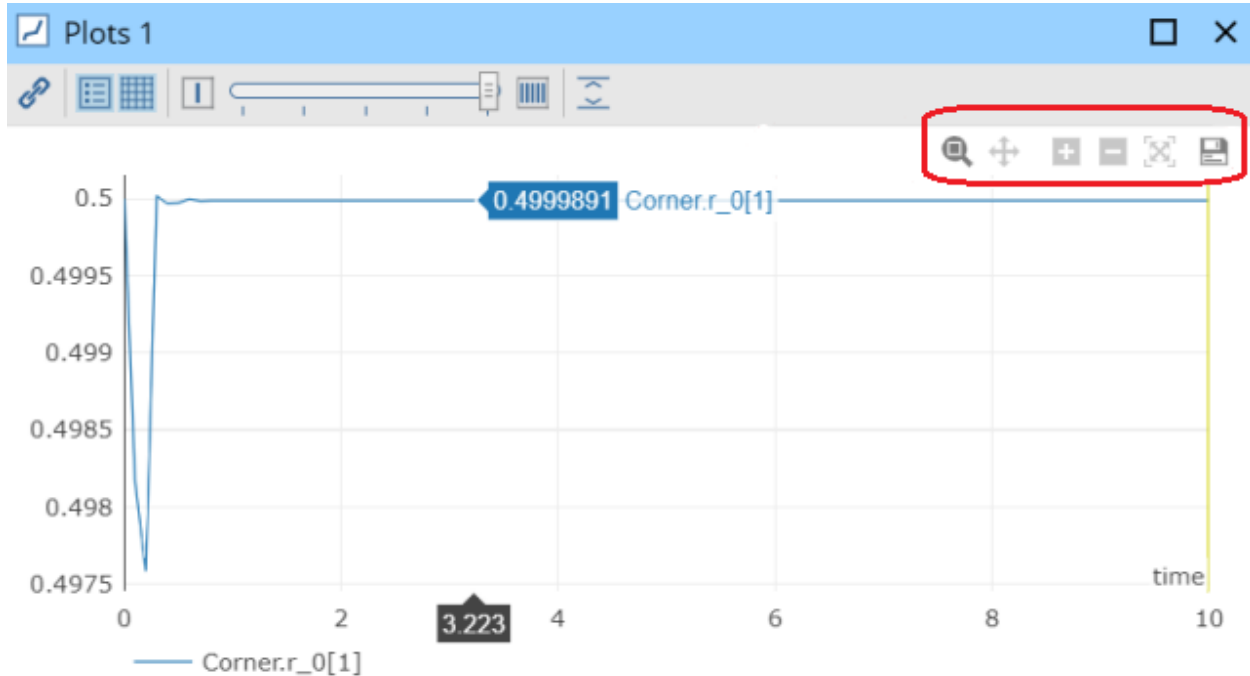
The following tools are displayed in the plot toolbar.

Control	Description
🔍	Zoom.
✛	Pan.
➕ ➖	Zoom in and out.
🗙	Autoscale. Resets view.
💾	Save plot data. Saves data to a comma-separated values file (.csv).

Use the pop-up toolbar to zoom, pan, zoom in and out, and autoscale.

When in **Zoom** mode (🔍), drag the pointer across the plot region to select a specific region to zoom in on. The selected region fills the plotting area. To return a plot to the original view, click **Autoscale** (🗙). Alternatively, double-click in a plot to return the plot to the original view.

When in **Zoom** mode (🔍), you can pan by holding **Shift** and dragging the pointer across the plot. If you drag along the x- or y-axis, you can pan only right/left or up/down.

For information on how to make a plot of multiple variables, see *The Variables Palette (page 13)*.

Tip: The yellow vertical line is the current time indicator. It can be toggled off using the **Show/hide time indicator** button (∿) in the main toolbar at the top of the MapleSim Insight application window. You can change the color of the time indicator line in Application Settings, in the Display Settings.

The Windows Palette

The **Windows** palette of the **Simulation Results** (📈) tab lists all the 3-D visualization windows and 2-D plot windows.

The following tools are displayed in the Windows palette.

Control	Description
3D	**Create new 3-D window.**
	Hide all windows. Hide all windows, making the visualization area empty.

By right-clicking in the Windows palette, you can also choose to show/hide the selected window, show or hide all windows, delete the selected window, or rename the selected window.

Windows that are not shown have their icons grayed out. Double-click on the name of a window in the Windows palette to toggle between show and hide window.

You can create up to four 3-D windows. There is no limit on the number of plot windows you can create.

The Variables Palette

The **Variables** palette of the **Simulation Results** ( ) tab lists inputs, outputs, parameters and variables for the current model. The palette is used to create new plots or to add a new variable to a plot. Use the search field in the Variables palette to find a variable quickly.
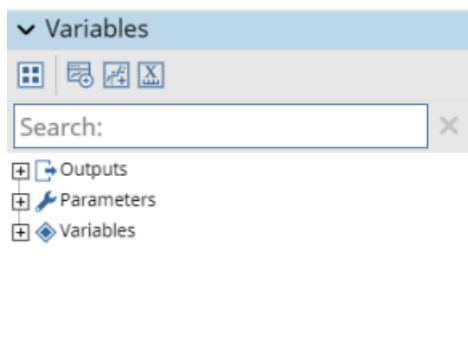
The following tools are displayed in the Variables palette.

Control	Description
	Add Variable to Traces. Select a variable then click this button to add a trace for this variable to the **Traces** window.
	Create new plot. Create a new plot for the selected variable.
	Add variable to existing plot. Add the selected variable to the selected plot.
	Set the x-axis variable. Place the selected variable on the x-axis of the selected plot.

You can create up to four 3-D windows. There is no limit on the number of plot windows you can create.

However, you can only have one traces window.

Right-click on a variable to access menu items to create new plot window, create new plot, add variable to plot, or set x-axis.

Double-clicking on a variable that is already plotted brings that plot window to the foreground with the plot for that variable selected, unless the **Traces** window is selected. In this case it will try to add the variable to the **Traces** window instead.

To create a new plot window configuration:

- Select a variable from the Variables palette. Click the **Create new plot** button (🖼). A new plot window is created containing the selected variable.
  Alternatively, if the variable is not yet plotted, double-clicking the variable from the Variables palette also creates a new plot window with this variable plotted in it.

To add a plot to an existing plot window:

1. In the Windows palette, select the plot window to which you want to add a plot.

2. Do one of the following:

- Select a variable from the Variables palette. Click the **Add variable to existing plot** button (📊).

- Double-click a variable from the Variables palette.

The new plot is appended to the end of the current plot window.

To add a variable to an existing plot:

- Select a plot, then right-click on a variable from the Variables palette and select **Add Variable to Plot**.
  Alternatively, select a variable from the **Variables** palette and drag it onto a plot.

By default, plots show time on the horizontal axis. To instead place a specified variable on the x-axis of a plot:

1. Select a plot.

2. Select a variable from the Variables palette.

3. Click the **Set the x-axis** button (📊).

To add a variable to the Traces window:

1. Select a variable from the Variables palette.

2. Click the Add Variable to Traces button (📊).

A trace plot of the variable is added to the Traces window.

Note: You can also select a variable from the tree view, and drag and drop it in the **Traces** or **Plots** window. If you drop it in the **Traces** window a new trace plot is created. If you drop it in the **Plots** window a new curve is added to the plot where you dropped the variable.

Plotting Diagnostic Variables

If you generate an FMU using MapleSim with the FMU generation app , the FMU will contain a feature that provides capability for you to plot diagnostic information.

When simulating this FMU from MapleSim Insight, there are currently five diagnostic variables that can be plotted:

inputLatency - A measure (in seconds) for how long it took the FMU to receive input before it could run a "step".
numStepsAdvanced - The number of (internal) steps that were advanced during the "step" calculation. If the FMU is being controlled by MapleSim/MapleSim Insight, then the **numStepsAdvanced** will be 1 + num internal steps, as specified in the FMU generation app, but only if the I/O interval is set to 1. If the I/O interval is set to 3 then it will be 3*(1+num internal steps).

If the FMU is being controlled by a third party then Insight does not control how to perform the steps to advance the FMU. In this case, Insight will only show what the third party software has done in the plot.

simulationTime - The current time of the simulation.

timeBetweenStepCalls - The time (in seconds) that has expired between individual "step" calls.

timeToComputeStep - The amount of time (in seconds) that it took to compute a "step". If this number is less than **FMU Step Size** then the FMU can run in real time or faster (usually if the input latency or anything else in the control loop isn't consuming a lot of time). If this value is larger than **FMU Step Size** then reduce the Sync Factor (less than 1) or else you will over flow the storage buffers and you will be disconnected from the running FMU.

Note: All of the above descriptions refer to a "step". Basically, one "step" means one data point will be captured. Internal steps are controlled by the FMU generation app at the time the FMU is created.

Recording 3-D Visualization Results

To record a video of your simulation:

1. Select Record Video ( ).

2. By default, the entire simulation is recorded.

3. When prompted, select the directory where you want to save your simulation, assign an appropriate name for the file.

4. Choose the video save format mp4/webm,then

5. Click **Save**.

Changing Video Recorder Settings

You can adjust the video recording settings from the **Recorder** tab in the **Application Settings** window.

Managing Simulation Results

You can manage your saved results inside the Result Management dialog.

To open the Result Management window, click Manage Results on the top toolbar.

The Results Management Window

The table in the Results Management window contains information on saved and most recent simulation results.

Specifically, the table contains the following information for each result:

Name- The name of the result. By default, the most recent result is named "Latest Result". When you save a result, you can edit the name to one of your choosing. Notice that, in addition to the name of the result, the **Name** field includes the date and time of the simulation.

Size- The size of the file that contains the simulation result.

Start Time- The start time of the simulation as defined in Insight or in the EDS file. The default value is 0.

End Time- The end time of the simulation as defined in Insight or in the EDS file.

Controller- There are three possible values: Insight (Insight used to simulate the model), Unknown (external tool used to simulate the model) and EtherNet/IP.

Notes- Provides a text area where you can add information about a specific result.

Saving and Naming Results

To Save your results:

1. Click on Save Results on the top toolbar.
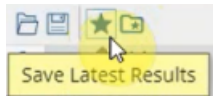
2. Your result is saved with a default name.

Alternatively,

1. Click on Manage Results on the top toolbar
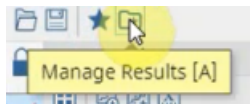
2. From the Result Management window, right-click on the result you want to save.

3. From the context menu select Save.

4. In the Rename window, enter a name for the saved result.
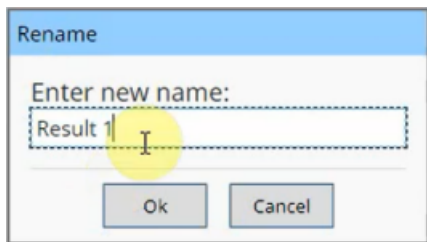
5. Click OK.

6. Click OK again in the Result Management window.

Notice that the corresponding plots in the plot window have the result name prepended to their names.

Adding Notes to Results

To add notes to a result:

1. In the Result Management window, right-click the result you want to add notes to.

2. From the context menu, select Edit Notes.

3. Add your notes to the text field in the Edit Notes window.

4. Click OK.

Working with Multiple Results

You can manage multiple simulation results from the Result Management window.

For instance, if you have a saved simulation result, you can modify values such parameter values, start time and end time back in the Insight Variables Palette. You can then save these results under a different name.

This process can be repeated for as many variations of the simulation as desired and the results will be available through the Result Management window.

Also, the output view in the Insight user interface will display, by default, these simulation results.

The plot view will display the plots of the simulation results data, grouping plots corresponding to the same result (same name prepended to the plot) together.

You can also choose to hide or display specific results in the user interface.

Hiding or Displaying Results

To hide a result:

1. Double-click the result you want to hide or display. If the display icon is disabled (👁) the result is hidden, if the icon is enabled (👁) the result is visible.

2. Click **OK**.

Alternatively,

1. Right-click the result you want to hide or display and select **Toggle Visibility**.

2. Click **OK**.

Application Settings

To open the Application Settings dialog, click **Application Settings** (⚙) in the main toolbar. The Application Settings dialog consists of the following tabs: Connection, Display, Values, Keyboard, Mouse, and About. Click a tab to view and customize the relevant settings.

Connection Settings

Control the connection settings. When using MapleSim Insight, the saved simulation data includes data from a window in time. You can control how much data is stored by adjusting these settings.

- Set the frame rate (in FPS) for the animation in the **Update Rate** field. By default the frame rate is 40 FPS. Note: The FPS is displayed in the FPS Indicator in the status bar, at the bottom right, of the Insight window.

- By default, **Keep All Data** is unchecked. The FMU will store the computed limit of **Samples per Second** and the user can change this value anywhere from 1 to the computed limit. This option is persisted across all FMUs.

- The **Samples per Second** limit that can be recorded for an FMU is limited by the defined step-size of an FMU (cannot be greater), or the amount of storage a single sample would require (larger models will have a smaller maximum number of samples per second).

- When **Keep All Data** is selected, the **Samples per Second** value will automatically be set to the computed limit and cannot be edited by the user.

- Set the **Max Buffer Size**. When the max buffer size is reached, the oldest data is thrown out so the latest captured data can be stored. The default is 250 MB. Note: The Max Buffer Size cannot be changed while a simulation is running (all other parameters can be changed).

Warning: If you decide to decrease the max buffer size (for example from 250 MB to 50 MB) data that is captured will be thrown out and cannot be recaptured without re-running your simulation.

- View the **Memory Usage**. This shows how much of the buffer has currently been used by the data that has been stored. This is also visible in the status bar at the bottom of the MapleSim Insight window.

- Set the **Timeout** value. This is the default timeout for communication between the automation tool and MapleSim Insight. When no data is transmitted for longer than this timeout, Insight automatically disconnects from the automation tool. In that case, you will not be able to see your results in MapleSim Insight. The default timeout is 3 seconds on Microsoft Windows, and 1 second on Linux. To override the default timeout value, you can adjust this option.

Display Settings

Control the display settings in 3-D visualizations.
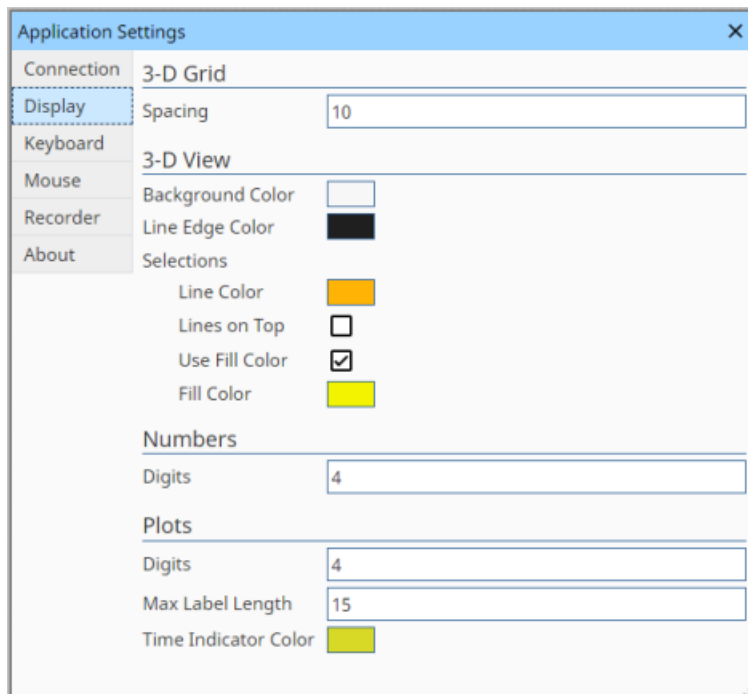
3-D Grid

- Set the grid **Spacing**. You can also access this setting in the 3-D Settings Dialog.

3-D View

- Specify a **Background Color** and **Line Edge Color** for 3-D visualizations.

- Under Selections, specify how selections are displayed. **Line Color** is the color used for the wireframe of the selected object(s). If **Lines on Top** is selected, the selection appears on top of other objects in the model. This option makes it easier to see the selection even if it is behind another object in the current view. If you select **Use Fill Color**, **Fill Color** is the color used for the filled body of the selected object.

Numbers

- Use the **Digits** field in this section to set the number of digits shown in the user interface (for example, the playback slider).

Plots

- Use the **Digits** field in this section to set the number digits displayed in plot axes or when you position your mouse on a plot line.

- Specify the **Time Indicator Color** for the time indicator line on 2-D plot windows.

- Use the **Max Label Length** option to control the maximum label length displayed in plots and traces.

Keyboard and Mouse Shortcuts

In the **Application Settings** dialog, you can review the current keyboard bindings and mouse settings. This page describes the default settings. The settings can be customized. The process is described below the table of defaults.

Keyboard Settings

Table 1: Default Keyboard Bindings

Action	Keyboard Shortcut
Arrange Windows - Cascade	**Shift + C**
Arrange Windows - Tile	**Shift + T**
Camera Mode Pan	**F2**
Camera Mode Rotate	**F4**
Camera Mode Zoom	**F3**
Camera Pose Restore	**R**
Camera Pose Save	**S**
Fit Plots To Window	**Shift + F**
Focus on Selected	**F**
Help	**F1**
Pan Camera Down	**Arrow Down**
Pan Camera Left	**Arrow Left**
Pan Camera Right	**Arrow Right**
Pan Camera Up	**Arrow Up**
Rendering Mode	**W**
Set Camera View along -X axis	**X**
Set Camera View along -Y axis	**Y**
Set Camera View along -Z axis	**Z**
Set Camera View along X axis	**Shift + X**
Set Camera View along Y axis	**Shift + Y**
Set Camera View along Z axis	**Shift + Z**
Show/Hide Grid	**D**
Show/Hide Orientation Box	**B**
Toggle Camera Type	**M**
View All	**V**

To change a keyboard binding:

1. Select a command from the list under **Application Settings > Keyboard**.

2. Enter the desired key or key combination. (Note that you can select multiple buttons, for example **Ctrl + A**.)
   If that key binding is already in use, the existing assignment is listed under conflicts. If it conflicts with an existing binding that cannot be changed, the conflict is listed in red. (For example, **Ctrl + C** is used for Copy and cannot be used for anything else.) If a conflict is displayed in black, it indicates the current assignment, but it can be reassigned.

3. Click **Assign** to make the change.

To remove a keyboard binding:

1. Select a command from the list under Application Settings > Keyboard.

2. Click **Remove** to remove the keyboard binding from that command.

Mouse Settings

Mouse Wheel Settings

- Roll the wheel to zoom 3-D View: If selected, rotating the wheel button zooms the 3-D view.

- Invert wheel zoom motion: This inverts the direction of the zoom.

Mouse Motion Settings

Table 2: Shortcuts for Pan, Zoom, and Rotate actions

Action	Mouse shortcuts
Pan 3-D View	**Right Button**
Rotate 3-D View	**Left Button**
Zoom 3-D View	**Middle Button**

To change the binding for a Mouse action:

1. Select a command from the list under **Mouse Motion Settings**.

2. Enter the desired key or key combination. (Note that you can select multiple buttons, for example **Ctrl** + **Shift** + **Left Button**.)
   If that key binding is already in use, the existing assignment is listed under conflicts. If it conflicts with an existing binding that cannot be changed, the conflict is listed in red. If a conflict is displayed in black, it indicates the current assignment, but it can be reassigned.

3. Click **Assign** to make the change.

Returning to Default Bindings

Settings changes are applied globally, so any changes apply to the current and future sessions of MapleSim Insight. If for some reason you need to revert to the default bindings, you can delete the files that store these settings. The next time MapleSim Insight is opened, the default bindings for the shortcuts will be restored. The files are

> **C:\Users\<name>\AppData\Roaming\MapleSimInsight\<version>\commands.json**

> **C:\Users\<name>\AppData\Roaming\MapleSimInsight\<version>\userPreference.json**

Recorder Settings

You can adjust the video recording settings from the **Recorder** tab in the **Application Settings** window.
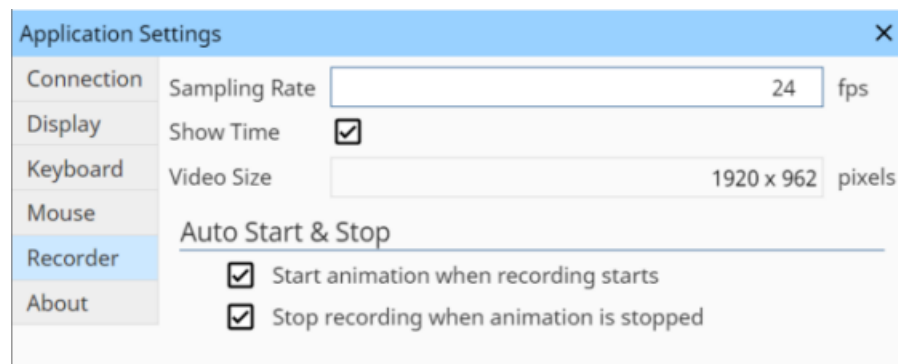
From this tab, you can change the Sampling Rate, display or hide the simulation run time, and decide when to start and stop the animation.

Note: The MapleSim Insight window controls the size of the simulation output window. So, to resize the video adjust the size of the MapleSim Insight window.

About MapleSim Insight

Under Application Settings, the About tab displays information about the current version of MapleSim Insight, including version, build ID, licensing information, and copyright information.

Running FMU Simulations from Maple

You can run FMUs in Insight from Maple using the Insight package for Maple. Specifically:

- CoSimulation FMUs created in MapleSim with the option, **Insight data** can be run directly.
- CoSimulation FMUs created in other tools (or in MapleSim with the **Insight data** option disabled) can be updated to a form that can be executed in Insight.

When you install Insight, if you have Maple installed, the Insight package for Maple will be installed for you.

Insight for Maple commands are of the form:

Insight:-Command(...)

While the FMU is running, Maple will launch Insight and will function just as if Insight was running the FMU directly.

Working with the EtherNet/IP Adapter

You can connect MapleSim Insight via EtherNet/IP to link your controller code to the inputs/outputs of your virtual machine for use with tools such as CODESYS or Rockwell Studio 5000 Environment.

The content below provides general information on EtherNet/IP Adapter options. For a detailed tutorial, including setting up the ethernet connection between the remote computer and the computer running MapleSim Insight, see *Connecting Insight to CODESYS Using EtherNet/IP (page 27)*.

Note: To use the EtherNet/IP feature, you need to launch MapleSim Insight as an administrator.
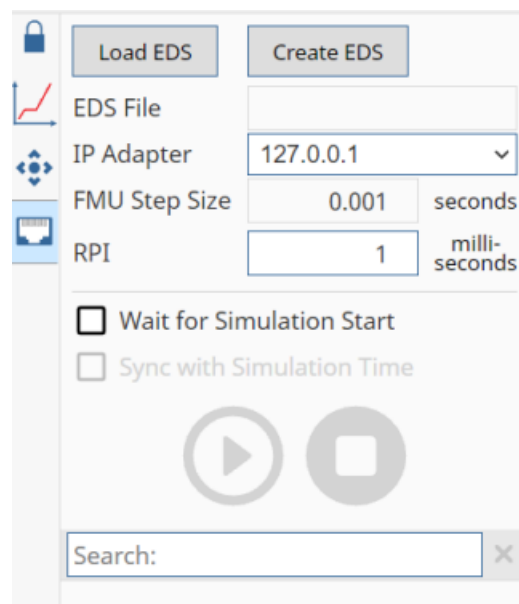
Load EDS Load an existing EDS file.

Notice the appearance of a new section, **EtherNet/IP Outputs**, which provides to access to these variables:

- `fmuInputDeltaTime` (time difference between the current and last data point received by Insight from the remote computer)

- `fmuInputTime` (input time values that were sent by the remote computer to Insight. This output is used for syncing the Insight controller with the remote computer)

- `timeouts` (a value of 0 or 1 indicating if/when a timeout occurred)
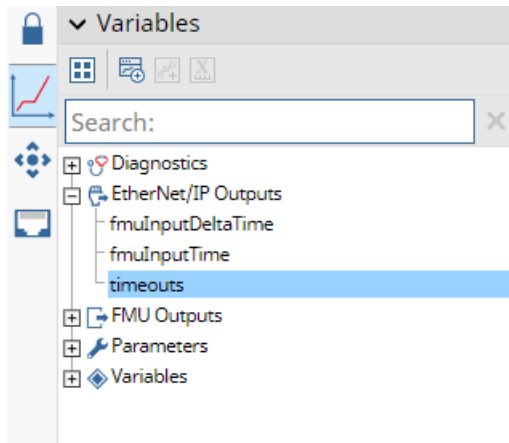
Create EDS Create an EDS file.

When you select Create EDS File, the Create EDS File dialog opens.
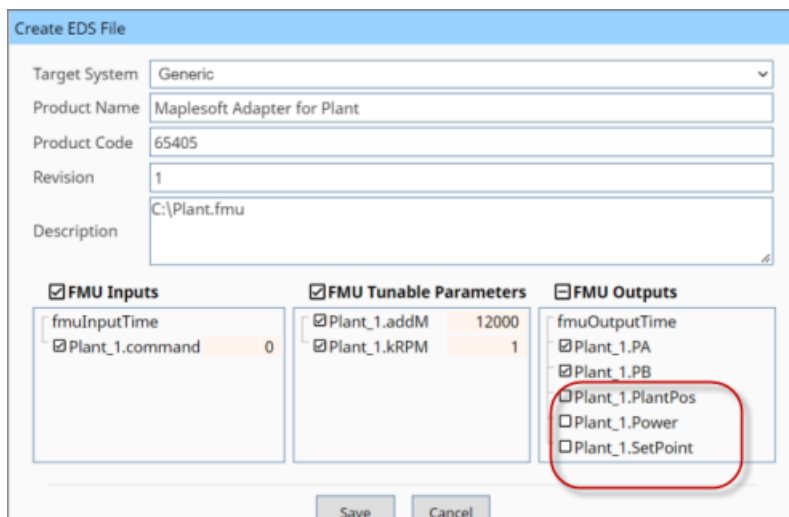
From here you can see that the FMU input variable is called fmuInputTime and the FMU output variable is called fmuOutputTime. Notice the checked values. These are values that are transmitted using EtherNet/IP. To select or de-select these values, double-click the variable.

Note: Values that are not checked are not transmitted using EtherNet/IP. However, they can be accessed in the parameters section of the EtherNet/IP tab, where they can be tuned.

Description of EtherNet/IP Tab Fields
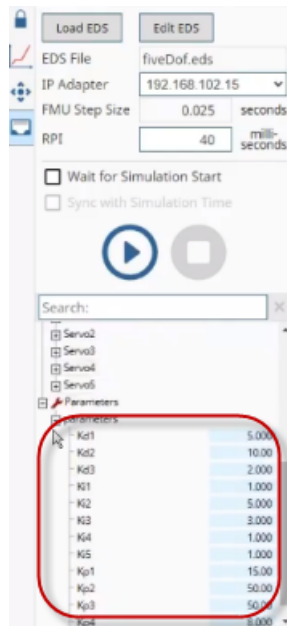
EDS File Text field that displays the name of the EDS file loaded into Insight. Moving your cursor over the text reveals the file location of this EDS file.

IP Adapter Choose the IP address of the computer that you wish to communicate with (In *Connecting Insight to CODESYS Using EtherNet/IP (page 27)*, this would be the IP address of the computer running CODESYS).

FMU Step Size This value is fixed. It is dependent on the FMU that you have loaded into Insight.

RPI The communication rate. This value can be tuned. By default it is the same value as the **FMU Step Size**, however you can tune it up or down. The minimum value allowed is one millisecond. The maximum value is one second.

Wait for Simulation Start If `fmuInputtime` is supplied by the other computer (for example, in *Connecting Insight to CODESYS Using EtherNet/IP (page 27)*, the computer running CODESYS) then you can check this box. This means that when you click the start button to begin the simulation, the simulation will not start until the remote computer starts sending signals corresponding to the `fmuInputTime` variable.

Sync with Simulation Time This checkbox is enabled when **Wait for Simulation Start** has been selected. If Sync with Simulation Time is checked, then if the remote computer is sending Insight a time signal then Insight will run the simulation at the rate that this time signal is indicating that the simulation should run at.

Description of Create EDS File Window Fields

Target System. Either Generic or Rockwell Automation. When you select Generic, an .eds file is generated which is sufficient to be viewed by most third party tools. When you select Rockwell Automation, an .l5x file is generated along with the .eds file. The .l5x file is necessary for import into Logix Designer® software.

Product Name. By default, "Maplesoft Adapter for" + <name of FMU>.

Product Code. An integer ranging from 65001 to 65535. This number is autogenerated based on the FMU file that you open and the location (file path) of that file. Note: External tools that load these FMUs will group things based on product code.

Revision. A floating point number, for your internal use, corresponding to your version numbering scheme.

Description. By default, the file path to the .FMU is displayed. You can also add other descriptive text as desired.

Troubleshooting

I cannot find a window.

Solution 1. Ensure the window is not hidden. Hidden windows have their icons grayed out in the Windows palette. Double-click the name of the window to toggle it to visible ( ).

Solution 2. The window may be minimized, behind another window, or have been moved out of the viewing area. For example, if you are working with the MapleSim Insight application maximized and then make your MapleSim Insight windows smaller, some plots may not be visible. To fix, maximize the MapleSim Insight application so that all plot windows can be found, and arrange them as needed.

Warning Message: "This FMU was created using a newer version of MapleSim. It may not display results as expected"

Solution 1. Regenerate the FMU in the version of MapleSim that corresponds to the version of Insight that you currently have.

Solution 2. Upgrade your version of MapleSim and Insight.

To upgrade your versions of MapleSim and Insight, contact sales https://www.maplesoft.com/contact/webforms/contact_sales.aspx

Connecting Insight to CODESYS Using EtherNet/IP

Introduction

MapleSim Insight includes functionality that allows the FMUs to communicate via EtherNet/IP. In this document, you are going to create such a communication using Insight and CODESYS. CODESYS is a development environment for programming controller applications. It is free of charge and can be installed legally without copy protection. Also, you are going to focus on a model of an injection molding machine as the main example. The generated FMU from this model will represent the physical model of the machine and the hydraulic circuit while the controller will be developed in CODESYS.

Note that these instructions are for users on a Windows platform.

CODESYS V3.5 SP16 Patch 1 + (64-bit)

Copyright (c) 2017-2020 CODESYS Development GmbH
Copyright (c) 1994-2016 3S-Smart Software Solutions GmbH
All rights reserved

Version Info...   License Info...

Prerequisites

There is no specific hardware dependency. MapleSim Insight supports connecting to EtherNet/IP capable hardware or software.

You will need two separate machines, however. One computer must have MapleSim Insight (2021.2 or newer) installed and the other must have CODESYS installed. The computer with CODESYS installed is running an EtherNet/IP Scanner while the computer with Insight installed is running an EtherNet/IP Adapter. The two computers are linked with a standard Ethernet cable connected to their standard Ethernet cards.

Running Insight EtherNet/IP connector

Configuring the Network

In order to have setup working, you must do the following:

1. Connect the two computers with an Ethernet cable. If a computer does not have an Ethernet port, you can use a USB to Ethernet converter.

2. Configure the IP addresses of the two computers on this point-to-point network. The addresses must be static and according to the table below. To configure the IP address on each computer, browse to **Control Panel> Network and Sharing Center> Change adapter settings**.

3. Right-click on the Ethernet connection and select **Properties**.

4. From the items list, double-click on **Internet Protocol Version 4 (TCP/IPv4)**.

5. Select **Use the following IP addresses**:

	CODESYS Computer	Insight Computer
IP Address	192.168.2.10	192.168.2.11
Subnet Mask	255.255.255.0	255.255.255.0
Default Gateway		192.168.2.10

Note: You can use any other IP addresses but you will need to use these IP addresses in later steps in this tutorial.

Testing the Network Setup

1. From the computer running CODESYS, open the Command window and run the `ipconfig` command. The output should be as follows (except for the `Link-local IPv6 Address`)

```
Ethernet adapter Ethernet:

   Connection-specific DNS Suffix  . :
   Link-local IPv6 Address . . . . . : fe80::a8a4:4494:70d0:bfed%14
   IPv4 Address. . . . . . . . . . . : 192.168.2.10
   Subnet Mask . . . . . . . . . . . : 255.255.255.0
   Default Gateway . . . . . . . . . :
```

2. On the same window, run `ping 192.168.2.11`. This will verify the IP-level connectivity by sending messages to the CODESYS computer and displays the receipt of the corresponding echo replies. This output should be similar to the following.

```
C:\Users\sbonab>ping 192.168.2.11

Pinging 192.168.2.11 with 32 bytes of data:
Reply from 192.168.2.11: bytes=32 time<1ms TTL=128
Reply from 192.168.2.11: bytes=32 time<1ms TTL=128
Reply from 192.168.2.11: bytes=32 time<1ms TTL=128
Reply from 192.168.2.11: bytes=32 time<1ms TTL=128

Ping statistics for 192.168.2.11:
    Packets: Sent = 4, Received = 4, Lost = 0 (0% loss),
Approximate round trip times in milli-seconds:
    Minimum = 0ms, Maximum = 0ms, Average = 0ms
```

3. On the Insight computer, run `ipconfig` command and verify the results.

4. On the same window, run `ping 192.168.2.10` command and verify the results.

If the `ping` command was not successful, it means that the Ethernet connection between the two computers is not established.

Create the EDS File

Electronic Data Sheets (EDS) are simply files that describe how a device can be used on an EtherNet/IP network. It describes the objects, attributes, and services available in the device.
You will need the EDS file for setting up the EtherNet/IP communication between Insight and CODESYS.

To create this file, follow these steps:

1. Open MapleSim Insight.

2. Load the FMU file of the injection molding machine, `Plant.fmu'.

3. Go to the **EtherNet/IP** tab.

4. Click **Create EDS**.

5. Change the **Product Name** to Plant and leave the other settings as default.

6. Click **Save** and save the file to your desired path.
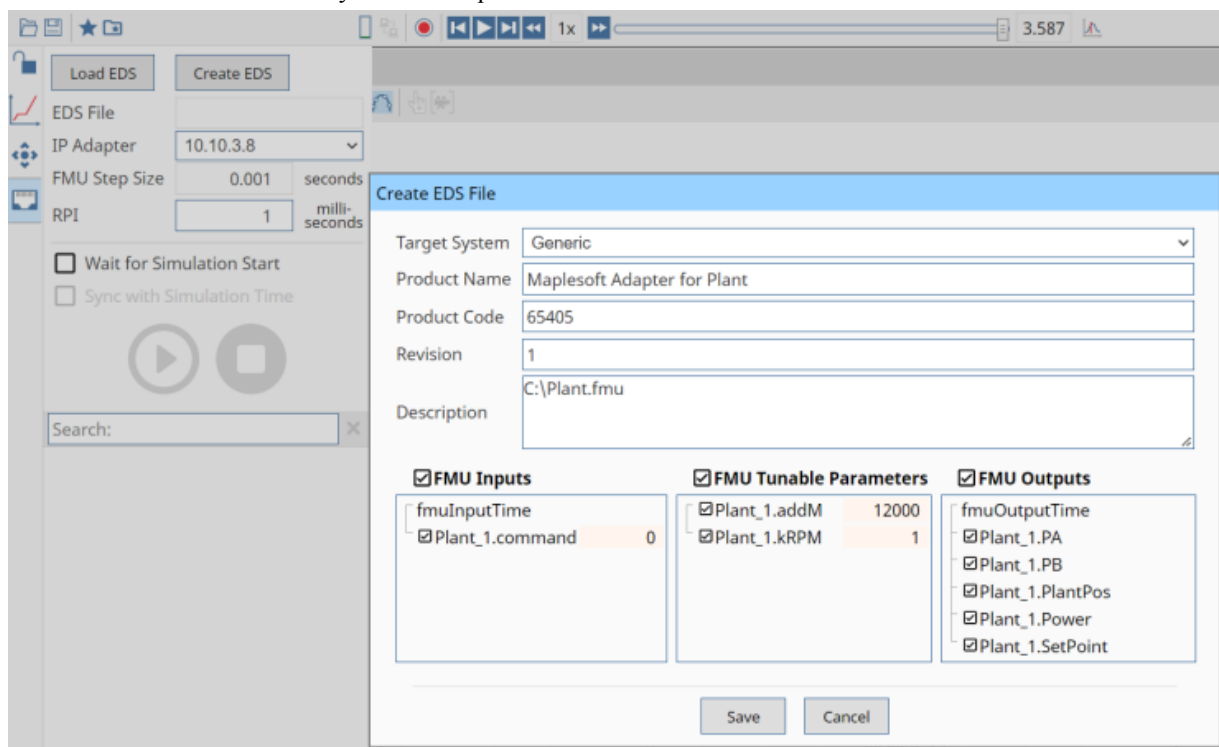
7. Close MapleSim Insight.
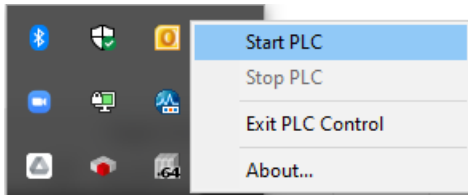
Setting up CODESYS

Configure CODESYS

The purpose of this section is to add an Ethernet device to the existing CODESYS project. You can find the initial CODESYS project file, **InjectionMolding Initial.project**, in the **Files** folder. Open this file in the computer that has CODESYS installed on it. This project includes a `PLC PRG' program that is going to be executed in cyclic when the project is being run. In this program, a PID controller is implemented using the continuous function chart (CFC) editor. The PID controller is going to control the motion of the center plate on the injection molding machine.
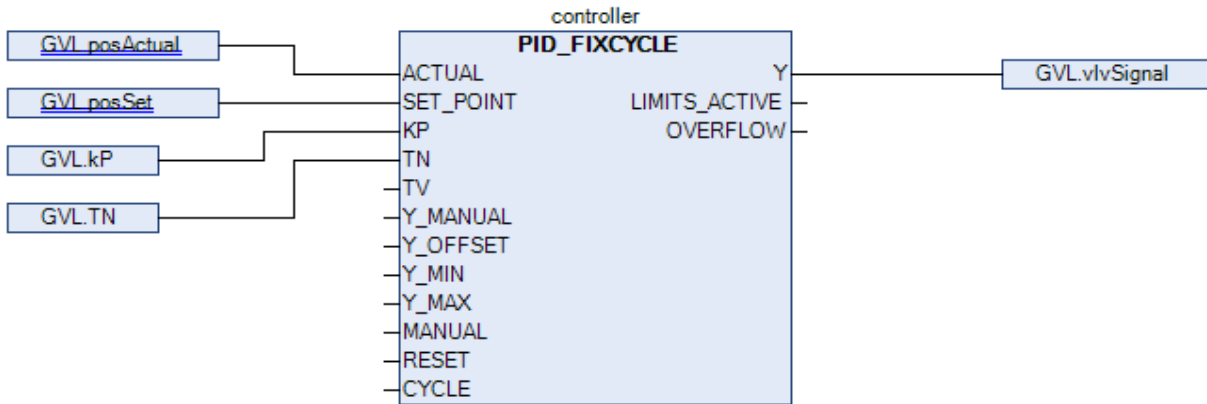
Follow the steps below to complete this project:

1. Show hidden icons in the System Tray from the Windows taskbar. Click on the **CODESYS Control Win Sys Tray** and, from the menu options, click **Start PLC.**


2. On the left panel, **Devices**, double-click on **Device (CODESYS Control Win V3)**. In the open window, from the **Communication Settings** tab, select your current machine as the **Device**.
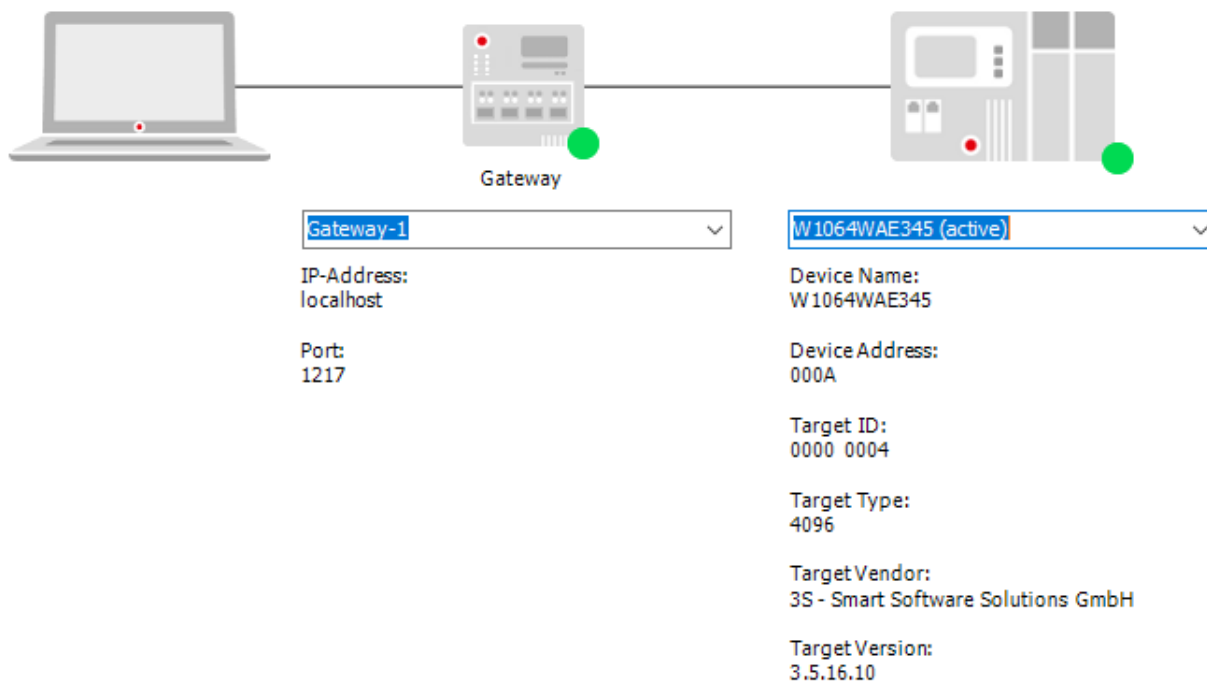
In this step, you are going to install an EtherNet/IP Adapter using the EDS file, **Plant.eds**. You generated this file in the previous section.

1. Go to **Tools> Device Repository** .

2. Click Install, and browse to the **Plant.eds** file and select it. You should be able to see the installed device in **Fieldbuses> EtherNet/IP> EtherNet/IP Remote Adapter** .
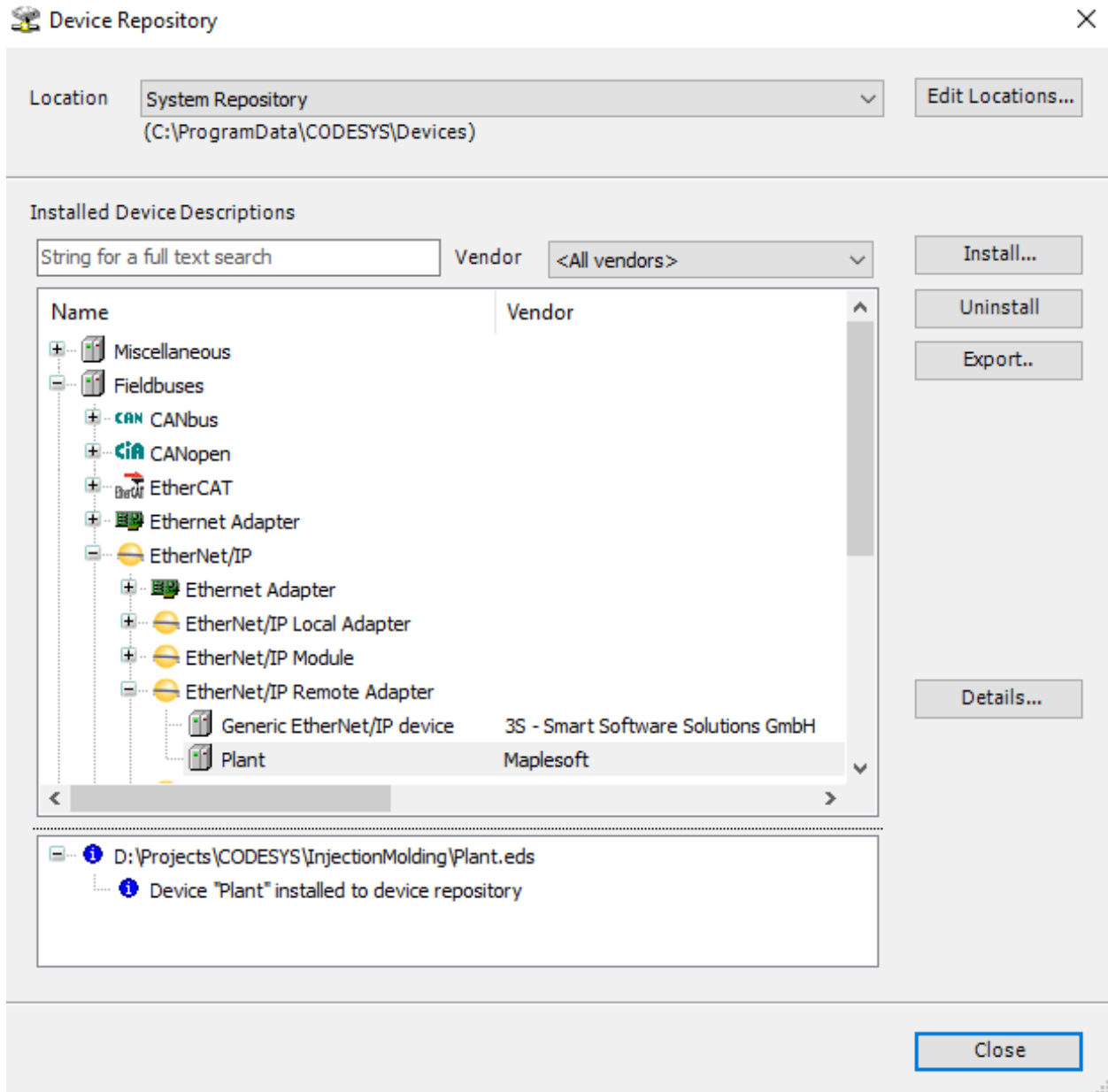
In this step, you need to add the installed device to the project and configure it.

1. Right-click on **Device (CODESYS Control Win V3)** and select **Add Device**.
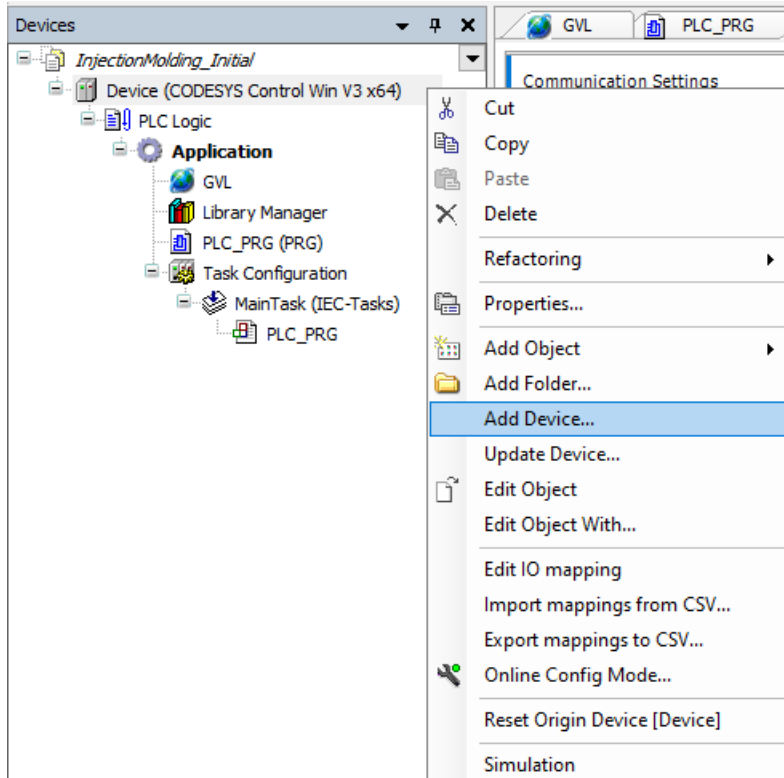
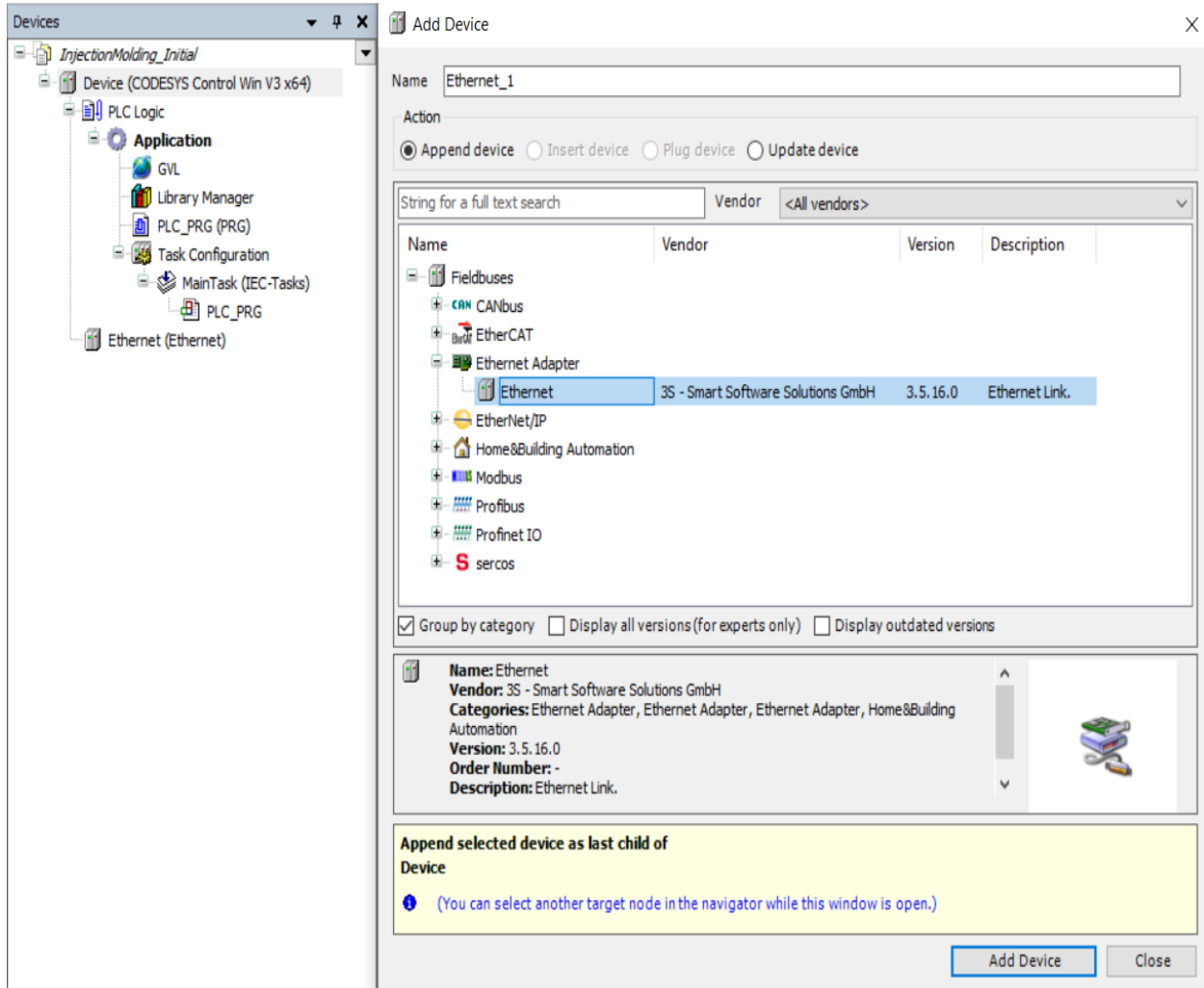2. Double-click on **Fieldbuses> Ethernet Adapter> Ethernet** to add an Ethernet device to the project. Notice how it appears on the **Devices** view on the left panel.

3. Keep the **Add Device** window open. Click on the newly added `Ethernet(Ethernet)' device. Notice how contents of the `Add Device' window changes. Double-click on

Fieldbuses> EtherNet/IP> EtherNet/IP Scanner> EtherNet/IP Scanner to add an EtherNet/IP Scanner device.
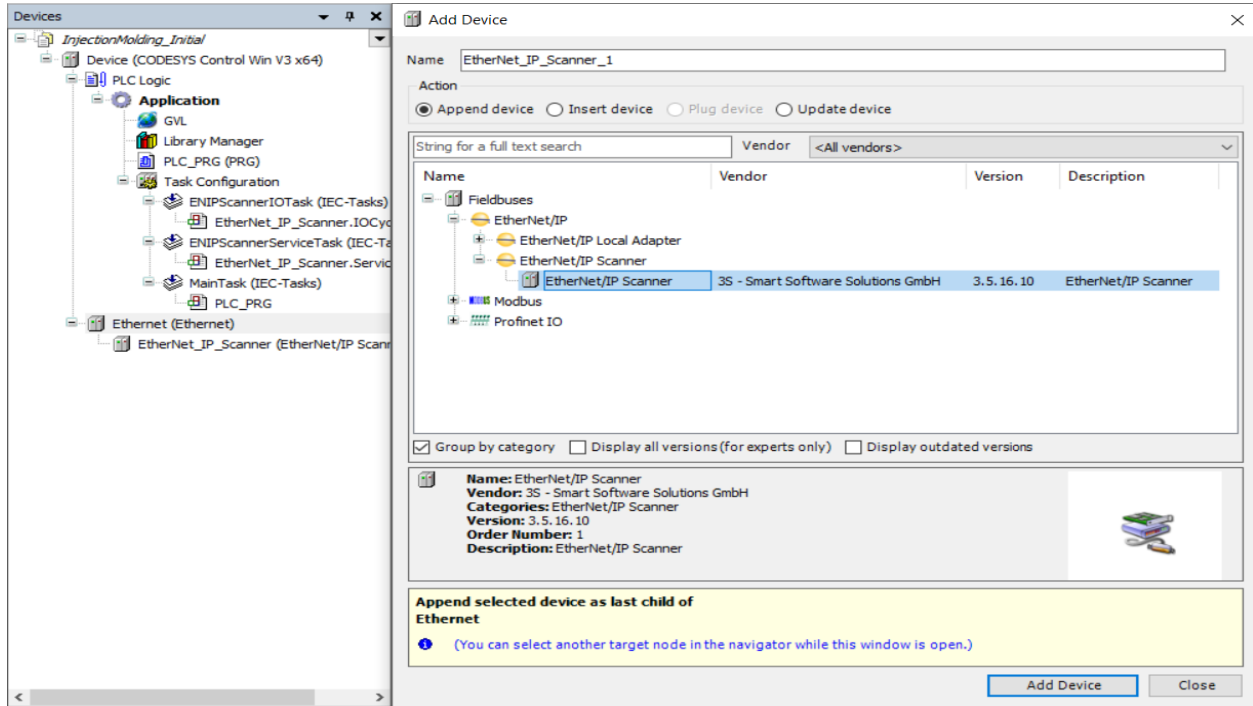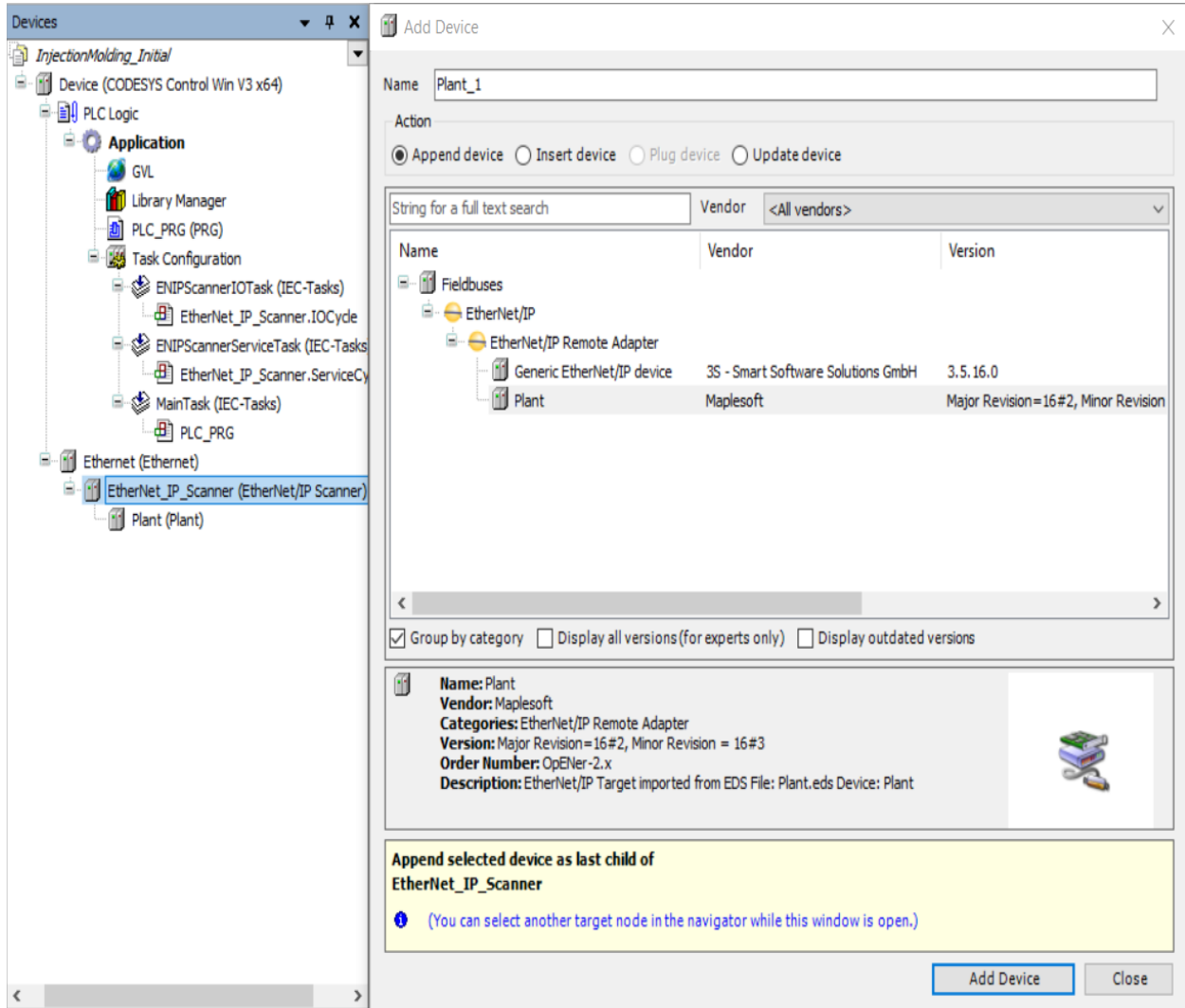
4. Keep the `Add Device' window open.

5. Click on the newly added `EtherNet IP Scanner' device. Notice how the contents of the `Add Device' window change.

6. Double-click on **Fieldbuses>EtherNet/IP> EtherNet/IP Remote Adapter> Plant** to add the Plant device.

7. Close the `Add Device' window.

At this point, you need to configure the recently added `Plant' device.

1. Double-click on the `Ethernet(Ethernet)' device.

2. In the `General' settings tab, click on ( ... ) and choose your Ethernet port as the `Network interface'.
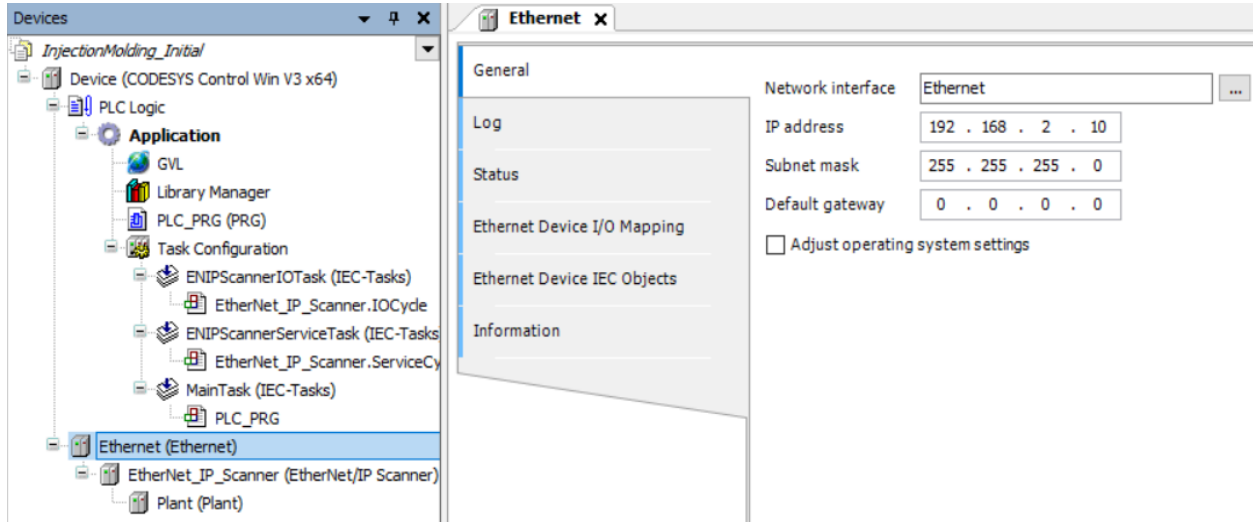
3. Double-click on the `Plant(Plant)' device.

4. In the `General' settings tab, change the `Address Settings' to 192.168.2.11
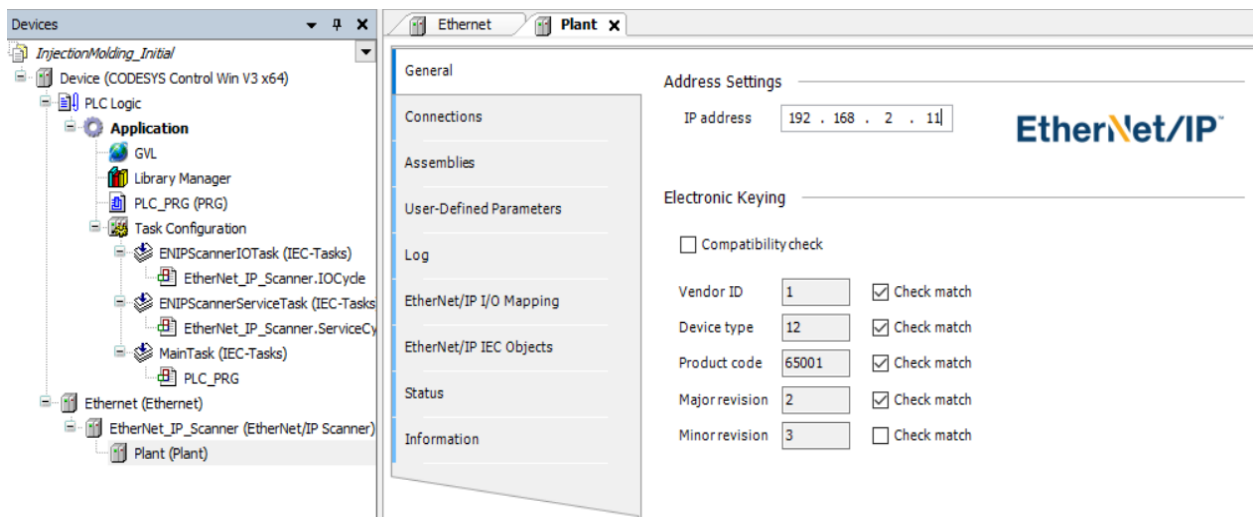
5. In the `EtherNet/IP I/O Mapping' tab, expand the `Exclusive Owner' and assign the I/Os as specified in the image below.

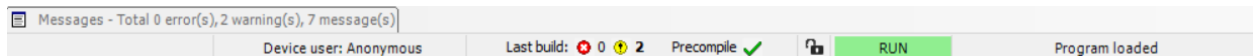| Variable		Mapping	Channel	Address	Type	Unit	Description
⊟ 📁 Exlusive Owner							
	🔷		fmuSimulationRunTime	%IL0	LREAL		fmuSimulationRunTime
	🔷		Main.Plant_1.PA	%IL1	LREAL		Main.Plant_1.PA
	🔷		Main.Plant_1.PB	%IL2	LREAL		Main.Plant_1.PB
🔷 Application.GVL.posActual		🔷	Main.Plant_1.PlantPos	%IL3	LREAL		Main.Plant_1.PlantPos
	🔷		Main.Plant_1.Power	%IL4	LREAL		Main.Plant_1.Power
🔷 Application.GVL.posSet		🔷	Main.Plant_1.SetPoint	%IL5	LREAL		Main.Plant_1.SetPoint
	🔷		fmuSimulationTargetTime	%QL0	LREAL		fmuSimulationTargetTime
🔷 Application.GVL.vlvSignal		🔷	Main.Plant_1.command	%QL1	LREAL		Main.Plant_1.command

Find — Filter Show all — Add FB for IO Channel... Go to Ins
General
Connections
Assemblies
User-Defined Parameters
Log
EtherNet/IP I/O Mapping
EtherNet/IP IEC Objects
Status
Information

6. At this point, the project is ready to be executed. Click (⚙️) on the toolbar and hit`Yes' in the confirmation window.

7. In toolbar, click (▶) to run the project. The status bar in the bottom should match the image below.

Messages - Total 0 error(s), 2 warning(s), 7 message(s)

| Device user: Anonymous | Last build: ⊗ 0 ⚠ 2 | Precompile ✓ | 🔒 | RUN | Program loaded |

Setting up Insight

After setting up the CODESYS project, switch to the computer running MapleSim Insight. Follow the steps below.

1. Open MapleSim Insight.
Note: Use the `Run as administrator' option to open MapleSim Insight.
2. Load the FMU of the injection molding machine, **Plant.fmu**.
3. Go to the **EtherNet/IP** tab.
4. Click on **Load EDS** and load the **Plant.eds** file that you had created initially.
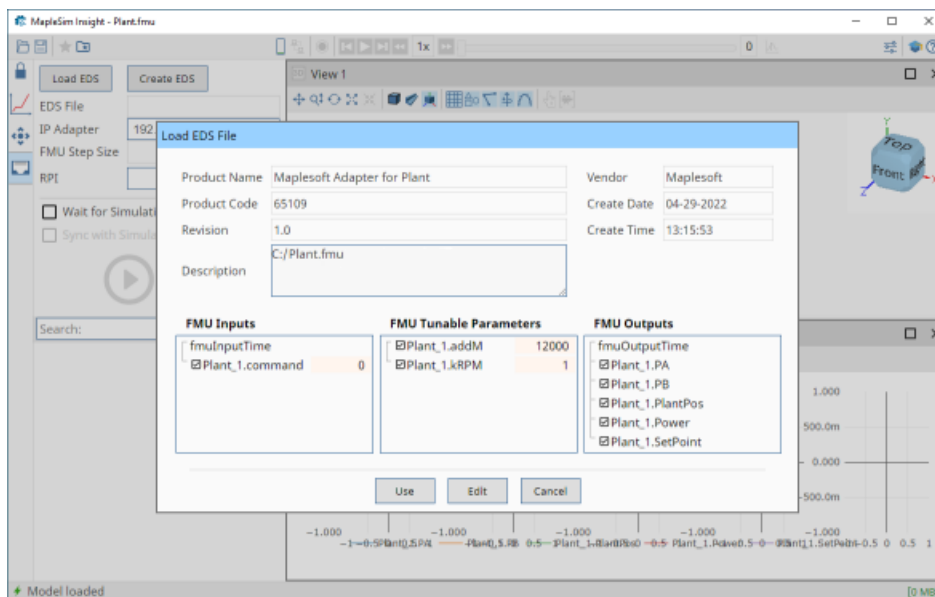
5. Leave all the fields as default and click **Use**.
6. Make sure that the `IP Adapter` is set to `192.168.2.11`.
7. Click on the (▶) button to run the FMU.

If the setup is configures correctly up to this point according to the discussed steps, you should be able to see the center plate on the injection molding machine in Insight being controlled by the PID controller in CODESYS. You can actually switch over to the `Master' controller and check the real-time value of the PID signals in the CODESYS project.
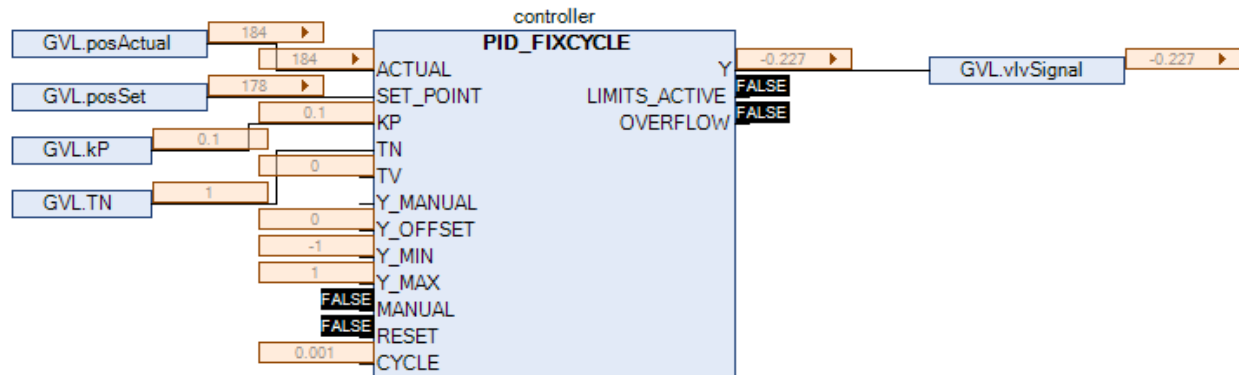
You can also change the PID gains and study the controller performance. For example

1. Open the window for the global variables by double-clicking on the **Device> PLC Logic> Application> GVL** .
2. In the prepared value column of the `kP' variable, enter 0.05.

Expression	Type	Value	Prepared value	Address	Comment
posSet	LREAL	37.9003...			Set position of the center platen [mm]
posActual	LREAL	76.5101...			Actual position of the center platen [mm]
vlvSignal	LREAL	0.21205...			Command signal to the DCV [-1..1]
kP	REAL	0.1	0.05		Proportional gain
TN	REAL	100	100		Integral reset time

3. Use `Ctrl+F7` to update the variable value.
4. Check the performance of the controller with the lower proportional gain.
You are done.

Legal Information

Maplesoft, Maple, and MapleSim are all trademarks of Waterloo Maple Inc.

© Maplesoft, a division of Waterloo Maple Inc. 1981-2022. All rights reserved.

No part of this document may be reproduced, stored in a retrieval system, or transcribed, in any form or by any means — electronic, mechanical, photocopying, recording, or otherwise. Information in this document is subject to change without notice and does not represent a commitment on the part of the vendor. The software described in this document is furnished under a license agreement and may be used or copied only in accordance with the agreement. It is against the law to copy the software on any medium except as specifically allowed in the agreement.

CODESYS is a registered trademark of the CODESYS Group.

Linux is a registered trademark of Linus Torvalds.

Studio 5000 Logix Designer is a registered trademark of Rockwell Automation, Inc.

Microsoft, Windows, Visual Studio and Visual C++ are registered trademarks of Microsoft Corporation.

All other trademarks are the property of their respective owners.

This document was produced using Maple and DocBook.

Third Party Software

License and copyright information for third party software distributed with MapleSim Insight is found in the file copyright.txt in your installation directory.